Purdue University
Purdue e-Pubs

International Compressor Engineering Conference | School of Mechanical Engineering

2016

Numerical Simulation of a Suction Valve: Comparison between a 3D Complete Model and a 1D Model

José Luiz Gasche
UNESP - State University of São Paulo, Department of Mechanical Engineering, Brazil, gasche@dem.feis.unesp.br

Allan Demétrio Sales de Lima Dias
UNESP - State University of São Paulo, Department of Mechanical Engineering, Brazil, allan.demetrio@gmail.com

Douglas Domingues Bueno
UNESP - State University of São Paulo, Department of Mechanical Engineering, Brazil, douglas@mat.feis.unesp.br

Jônatas Ferreira Lacerda
Tecumseh do Brasil, Research and Development Department, Brazil, jonatas.lacerda@tecumseh.com

Numerical Simulation of a Suction Valve: Comparison Between a 3D Complete Model and a 1D Model

José L. GASCHE[1]*, Allan D. S. L. DIAS[1], Douglas D. BUENO[1], Jônatas F. LACERDA[2]

[1]Unesp-State University of São Paulo, Department of Mechanical Engineering
Ilha Solteira, SP, Brazil
+55 18 3743 1035, gasche@dem.feis.unesp.br

[2]Tecumseh do Brasil LTDA, Research and Development Department,
São Carlos, SP, Brazil
+55 16 3633 7678, jonatas.lacerda@tecumseh.com

* Corresponding Author

ABSTRACT

Hermetic reciprocating compressors are widely used in small and medium size vapor compression refrigeration systems. One of the main parts of this type of compressor is the automatic valve system used to control the suction and discharge processes. As the opening and closing of the valves are caused by the forces produced by the refrigerant flow itself, the understanding of this fluid-structure interaction problem which characterizes the system dynamics is of fundamental importance in order to enhance the efficiency of the valve system. The numerical simulation of the flow has been used as an efficient method to perform this task. Most of the researchers simulated this problem by considering whether a geometric simplification for the valve (radial diffuser) or a simplification for the dynamics of the structure (one degree of freedom). In order to overcome these two drawbacks, in this work we simulate numerically the fluid structure interaction problem in a real geometry usually used in suction valves by considering a 3D complete model for both the flow and the dynamics of the structure. The Ansys CFX CFD-Computational Fluid Dynamics software was used for the simulation of the flow coupled with the structure module available in the Ansys Mechanical. In addition, a 1D model (one degree of freedom) was also implemented for simulating the dynamics of the structure for comparison reasons. The models were validated against experimental data for the aperture of the valve. We show that the 1D model saves small computational time and provides good results for general characteristics of the problem as the force acting on the valve and the aperture of the valve, while the 3D complete model allows us to predict the deformation of the structure. We suggest the use of the 3D complete model because the computational time is not significantly increased.

1. INTRODUCTION

Hermetic reciprocating compressors are widely used in small and medium size refrigeration cycles based on the vapor compression process. The thermodynamics losses (inlet superheating and flow through the suction and discharge systems) of high efficiency compressors can represent up to about 30% of the total losses - electrical, friction, thermodynamics, and cycle - (Possamai and Todescat, 2004). Among the thermodynamics losses, the suction and discharge flows through the suction and discharge valves can contribute with about 47% of the losses (Ribas *et al*., 2010). Therefore, the flow through the valve system can represent about 15% of the total losses in this type of compressor, which means that small efficiency increase of the suction and discharge processes can provide significant increase of the global performance of the compressor.

In order to improve the efficiency of the suction and discharge processes, it is essential to investigate the flow through the suction and discharge valves. As the movement of the valves (reed type valves) depends on the forces of the flow acting on its surfaces and on the structural response of the reed, this is a very complex problem because it involves the solution of the Fluid-Structure Interaction (FSI) between the refrigerant flow and the reed.

The FSI problem in refrigeration compressor valves has been extensively investigated in the last decade. Several numerical modeling dealing with the full three-dimensional FSI problem have been developed (Kin *et al*., 2008; Estruch *et al*., 2014; Silva and Arceno, 2014; Tan *et al*., 2014; Wu and Wang, 2014). Using this type of modeling we can obtain detailed characteristics of both fluid flow field and structural stress field, which can be very helpful for understanding the reasons of the thermodynamics losses. The main disadvantage of this modeling is the large computational cost. Using a hybrid modeling, in which whether the fluid flow or the structure dynamics is modeled in a simplified way (Kinjo *et al*., 2010; Mistry *et al*., 2012; Mayer *et al.*, 2014; Brancher and Deschamps, 2014; Yoshizumi *et al*., 2014; Ding and Gao, 2014), one can diminish the computational cost. However, we do not obtain detailed results and we always need other information based on experimental or numerical results in order to complete the models. For optimization applications, in which several design have to be evaluated, it would be important to compare both type of modeling in order to evaluate their computational costs as well as the accuracy of the results.

We present here a comparison between two methodologies developed to simulate the flow through a real geometry usually used in suction valves. In the first methodology we consider a complete three-dimensional solution for both fluid flow and structural dynamics, in which a Finite Volume Method (FVM) is used to solve the fluid flow, and a FEM is used to solve the structure behavior of the reed. For the second methodology we consider the same model for the fluid flow but a simplified model to solve the structural dynamics, in which a one-degree of freedom mass-spring-damping (1D model) system is used to model the reed. The complete three-dimensional model (3D model) is validated by using experimental results for the lift of the reed obtained from an experimental setup design to study compressor valves. The main objective of the work is to evaluate which methodology is more appropriate for optimization purposes in industrial applications.

2. METHODOLOGY

2.1 Numerical Methodology

A schematic view of the geometry of the problem (solution domain) is shown in Fig. 1(a), which is the model for the same configuration experimentally studied by Gasche *et al*. (2014). In the solution domain, there is an entrance region representing the inlet tube of the experimental setup (aluminum tube, Fig. 2a) and an outlet region representing the test section (upper plate and reed, Fig. 2b). The diameter ratio and thickness of the reed are D/d=1.3 and 0.4 mm, respectively. Two parameters that play an important role on the dynamic of the reed are the stiffness, k, and the natural frequency, f_n. These two parameters were obtained numerically by using the commercial code Ansys-Mechanical®, resulting in k=214.7 N/m and f_n=31.4 Hz.

Two physical models were developed. In the first model, a three-dimensional formulation was considered for both fluid flow and structural dynamics. The governing equations for transient, compressible and isothermal flow of a Newtonian fluid (air) consisted of the mass conservation equation, momentum equation and an equation of state relating the pressure and the specific mass of the fluid (ideal gas was used here). The boundary conditions shown in Fig. 2 are: prescribed average velocity at the inlet of the flow (inlet), no-slip condition at all solid walls (wall), and ambient pressure at the outlet of the flow (outlet). The system of equations was solved by using the Finite Volume Method (FVM) implemented in the commercial code Ansys-CFX®. After performing a mesh independence test, we adopted a mesh containing 137,000 volumes (Fig. 1 b and c). A deforming mesh (expansion/contraction) was used near the reed in order to capture its movement. Based on preliminary tests we chose the k-Ω Shear-Stress-Transport (SST) in order to model the turbulence phenomenon, and a high order resolution scheme was used to discretize the convective terms of the momentum equation. The governing equation for the structure (reed) is given by:

$$[M]\{\ddot{x}\} + [C]\{\dot{x}\} + [k]\{x\} = \{F\} \tag{1}$$

where $[M]$, $[C]$, and $[k]$ are the mass, damping, and stiffness matrices, respectively; x, $\dot{x}$, and $\ddot{x}$ are the local position, velocity, and acceleration vectors, respectively; and $\{F\}$ is the force vector of the flow acting on the reed surfaces. Equation (1) was solved by using the FEM implemented in the commercial code Ansys-Mechanical® for a 5,600 elements mesh. The structural system was based on quadratic tetrahedral elements with ten nodes and three degree of freedom per node. The contact between the reed and the seat was modeled as zero velocity for the reed. The damping matrix was considered to be proportional to the mass and stiffness matrices:

$$[C] = \alpha_d[M] + \beta_d[k], \tag{2}$$

where the proportionality constants are α_d=3.63 and β_d=7.703x10^{-6}.

The fluid-structure interaction problem was solved by using an uncoupled method, in which the fluid flow and the structural dynamics are solved separately using the latest data provided from each other.

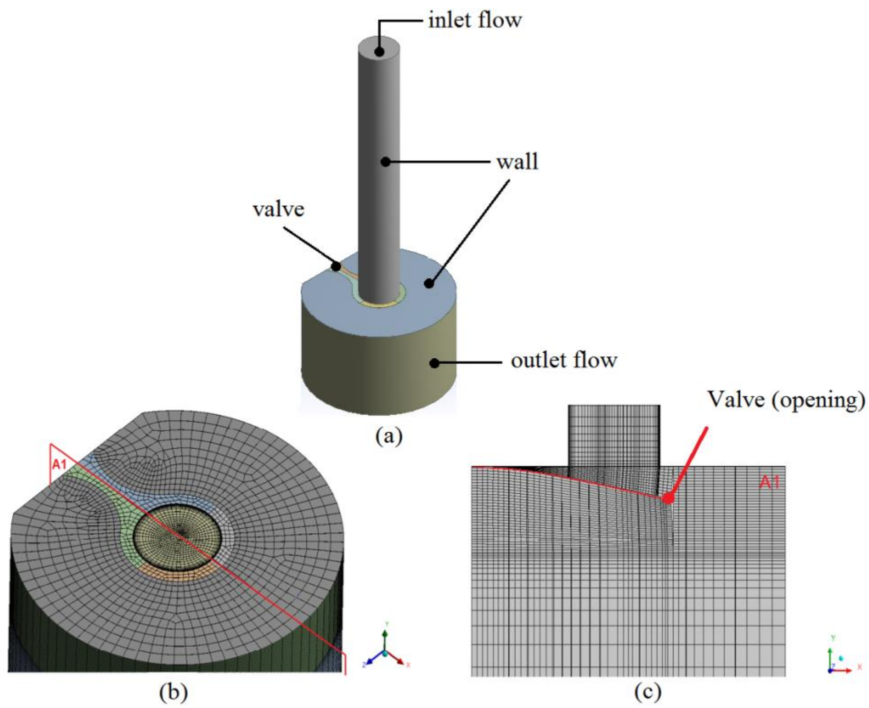


Figure 1: (a) schematic view of the solution domain; (b) top view of the mesh adopted for simulation; (c) longitudinal view of the mesh - plane A1

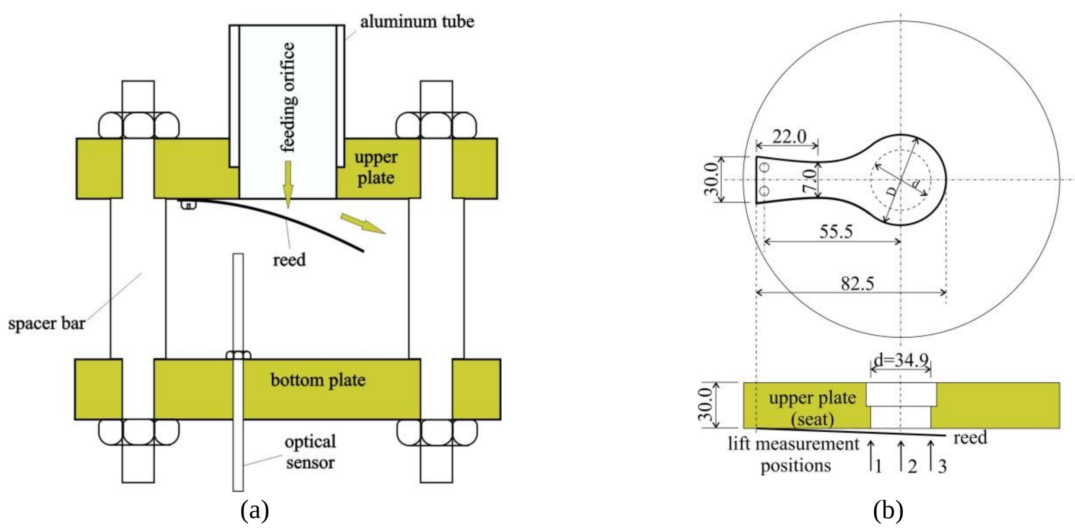


Figure 2: (a) test section; (b) upper plate and reed (Gasche *et al*., 2014)

For the second methodology, the fluid flow was solved by using the same procedure adopted in the first methodology. However, the structural dynamics was solved by considering a simplified one-dimensional (1D) model, in which the reed was taken as a non-deforming rotating beam with a concentrated mass-spring-damping at the center of the circular part of the reed (Fig. 3). For this case, the structural dynamics is governed by a one-degree of freedom mass-spring model described by Eq. (3).

$$m_T\ddot{y} + c_{eq}\dot{y} + k_{eq}y = F_{eq}, \tag{3}$$

where m_T is the total mass of the valve, y is the position of the valve, and c_{eq} and k_{eq} are the equivalent damping and stiffness coefficients of the reed, respectively. The resultant force, F_{eq}, acting on the reed surfaces is calculated by integrating the pressure field just on the circular region of the reed in order to avoid computational expenses to treat the transfer of data between the fluid and structure meshes. In addition, considering that we have to use a very small time interval, Δt, for integrating the equations governing the fluid flow, we used a constant force (determined at the preceding time) in order to solve Eq. (3), yielding to:

$$y = e^{-\alpha^*\Delta t}\left[y_c\cos(\omega^*\Delta t) + \frac{\alpha^* y_c - \dot{y}^0}{\omega^*} sen(\omega^*\Delta t)\right] + \frac{F_{eq}}{k_{eq}} \tag{4}$$

where,

$$\alpha^* = \frac{c_{eq}}{2m_T}, \qquad \omega^* = \frac{\sqrt{4m_T k_{eq} - c_{eq}^2}}{2m_T} \qquad y_c = y^0 - \frac{F_{eq}}{k_{eq}}, \tag{4a}$$

where y^0 and $\dot{y}^0$are the valve displacement and valve velocity at the preceding time; m_T is the total mass of the reed; and the equivalent stiffness coefficient, k_{eq}, was determined using m_T and the first natural frequency obtained by a numerical modal analysis. The definition of the equivalent damping coefficient c_{eq} was based on the same procedure used to determined matrix $[C]$. Different values were not evaluated once that experimental tests showed undamped motions. Equation (3) and the position of the reed at each time step were implemented in the CFX code by using a user defined subroutine.

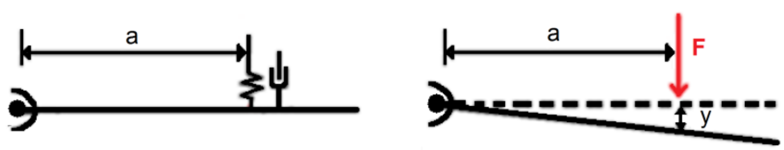


Figure 3: Schematic representation of the 1D structural dynamics

2.2 Validation of the Numerical Methodology

The 3D numerical methodology was validated against experimental data of the displacement of the reed for Reynolds number based on the inlet parameters of the flow (Re=Vd/ν) equal to 6,000. We measured the displacement of the reed at positions 1, 2, and 3 (Fig. 2b) using an optical sensor with measurement uncertainty equal to ±1μm. Details of the experimental apparatus and procedure can be seen in Gasche *et al*. (2014). The prediction of the reed displacement provided by the 3D model is qualitatively reasonable (Fig. 4). However, the model over-predicts the amplitude at the measurement positions 1 and 2 (Fig. 2b) by approximately 13% and 22%, respectively. In addition, the model over-predicts the frequency of the movement by a smaller scale, with a difference of approximately 5%. These differences may be due to the impact of the reed on the seat, which is not completely modeled here.

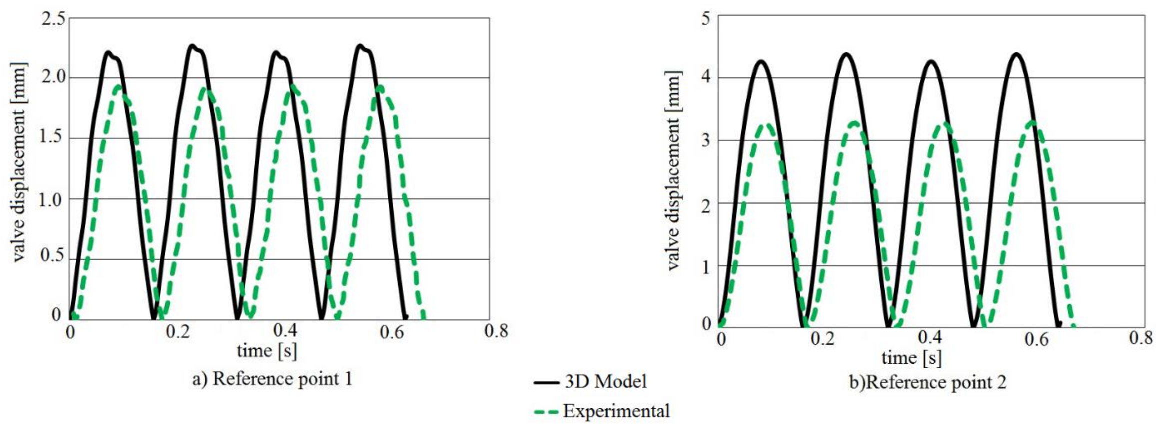


Figure 4: Displacement of the reed for Re=6,000

3. RESULTS AND DISCUSSIONS

In order to evaluate both methodologies (3D and 1D), we plotted the reed displacement and the resultant force (Fig. 5), and the pressure and velocity fields (Fig. 7 and Fig. 8) for Re=10,000. The best agreement for the valve displacement occurs at position 2. The disagreement at positions 1 and 3 is due to the deformation of the reed provided by the 3D model (Fig. 6). Figure 5d shows a very good agreement of the resultant force acting on the reed, except at the contact instant where a pressure pick is captured. Concerning the pressure and velocity fields (Fig. 7 and Fig.8), we do not see significant differences between the results provided by both models, which means that the simplified 1D model could be used without losing accuracy, mainly if global parameters as the resultant force and the mass flow rate (related to the valve aperture) are the concerns. Similar results were obtained for Re=4,000 and Re=6,000.

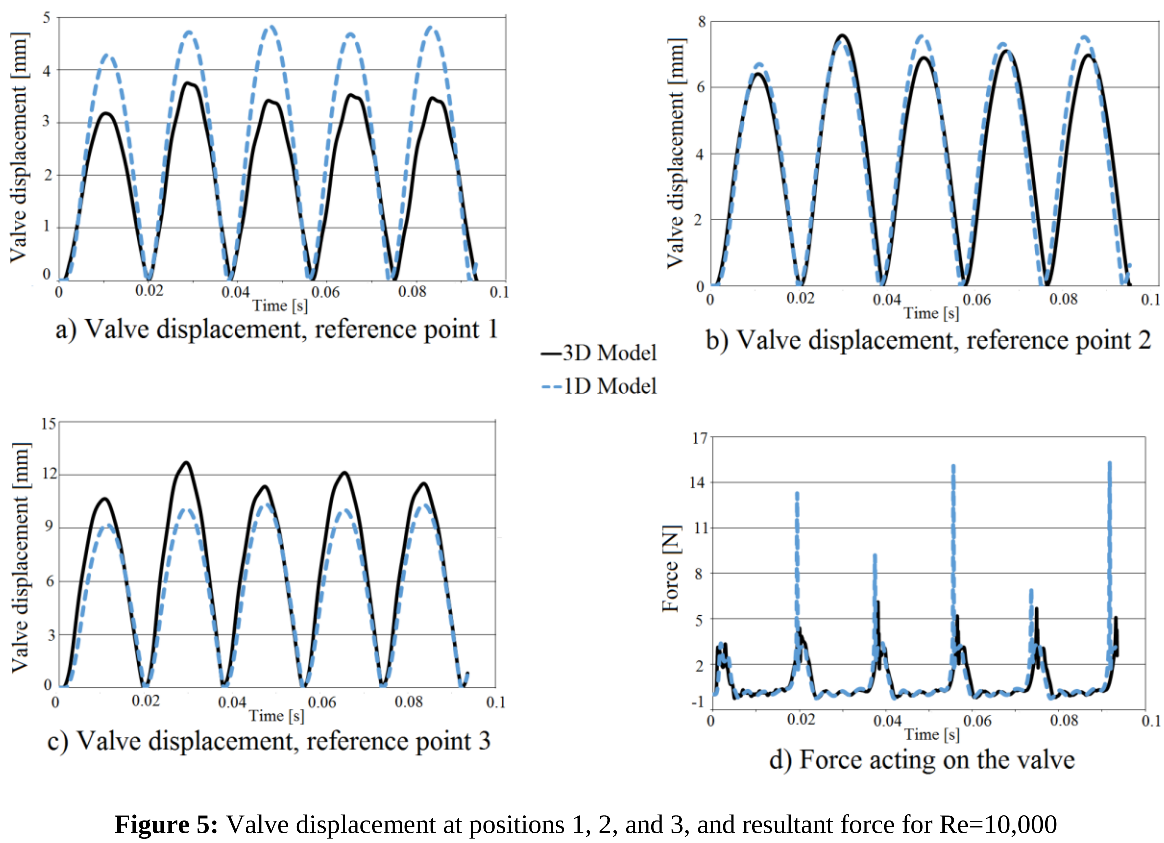


Figure 5: Valve displacement at positions 1, 2, and 3, and resultant force for Re=10,000

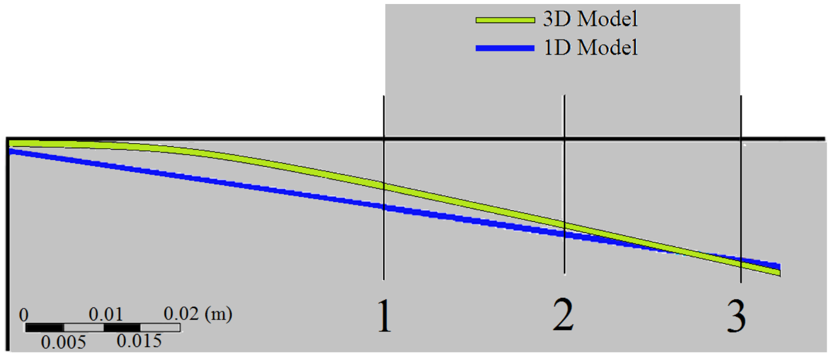
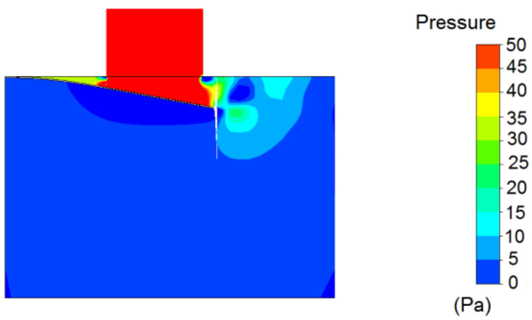
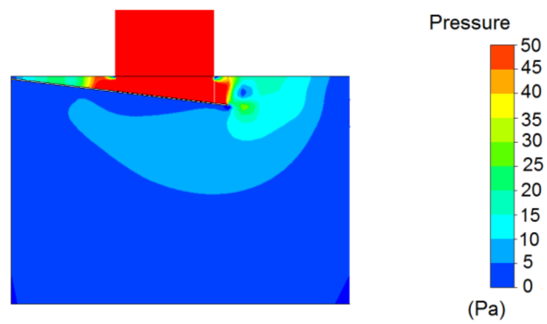


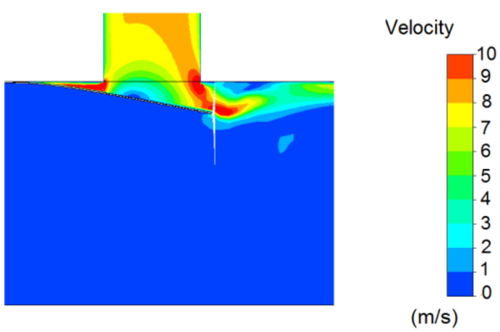
Figure 6: Positioning of the reed for the maximum aperture considering the 3D and 1D models



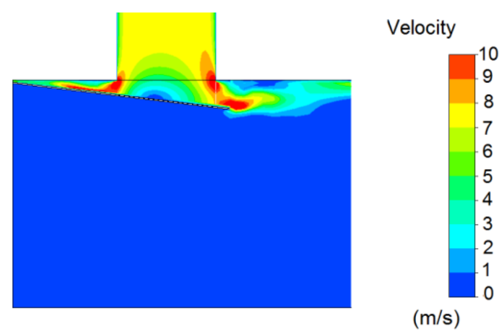
(a) Pressre field - 3D model



(b) Pressure field - 1D model

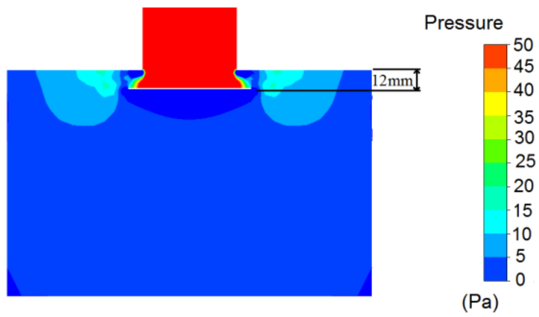


(c) Velocity field - 3D model

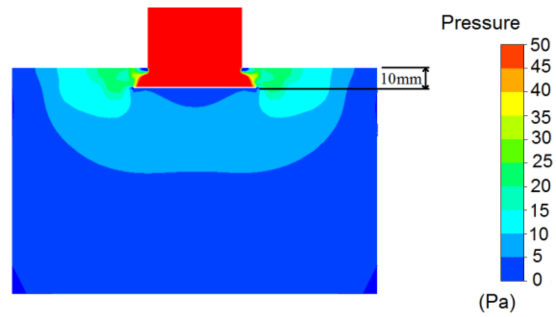


(d) Velocity field – 1D model

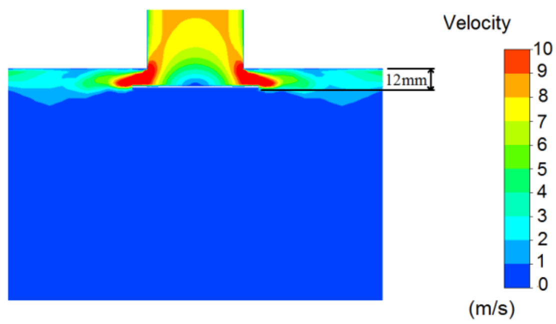
Figure 7: Pressure and velocity fields at the longitudinal view (plane A1) for Re=10,000



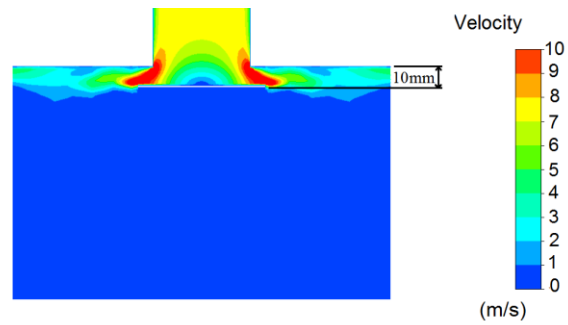
(a) Pressure field – 3D model



(b) Pressure field – 1D model



(c) Velocity field – 3D model



(d) Velocity field – 1D model

Figure 8: Pressure and velocity fields at the cross-section view for Re=10,000

We could indicate the 1D model for compressor simulation purposes if the computational time for running each simulation was much lower than that necessary to run the 3D model. However, we save just approximately 15% (2 hours) of the time needed to simulate the same case by using the complete 3D model (Table 1). For all cases we used the single core Intel® processor i7-3770, with 3.40 GHz and 16.0 GB RAM.

Table 1: Real time simulation of the problem

Re	3D Model	1D Model	Difference (%)
4,000	13h 32min	11h 40min	13.8
6,000	13h 37min	11h 42min	14.1
10,000	13h 59min	11h 48min	15.6

4. CONCLUSIONS

In this paper we evaluate the possibility of using a simplified 1D model over a complete 3D model in order to represent the structural dynamics of reed type valves. The objective is to assess the global parameters (force and displacement) and the real computational time spent to run both models. Concerning the global parameters there are not significant differences between the results. However, only 15% of reduction for the computational time does not justify the use of the 1D model, mainly because the 3D model can provide detailed results that are essential for design purposes as for instance the stress field in the valve.

REFERENCES

Brancher, R. D., Deschamps, C. J., (2014). Modeling of rolling-piston compressors with special attention to the suction and discharge processes. In *Proceedings of the 22nd International Compressor Engineering Conference at Purdue*, Paper 1302. West Lafayette-IN, EUA.

Ding, H., Gao, H., (2014). 3-D Transiente CFD model for rolling piston compressor with dynamic reed valve. In *Proceedings of the International Compressor Engineering Conference at Purdue*, Paper 1548. West Lafayette-IN, EUA.

Estruch, O., Lehmkuhl, O., Rigola, J., Pérez Segarra, C. D., (2014). Fluid-structure interaction of a reed type valve subjected to piston displacement. In *Proceedings of the 22nd International Compressor Engineering Conference at Purdue*, Paper 1659. West Lafayette-IN, EUA.

Gasche, J. L., Arantes, D. M., Andreotti, T., (2014). Experimental analysis of the fluid structure interaction in a suction valve model. In *Proceedings of the 22nd International Compressor Engineering Conference at Purdue*, Paper 1217. West Lafayette-IN, EUA.

Kim, H.S., Ahn, J.W., Kim, D.H., (2008). Fluid structure interaction and impact analyses of reciprocating compressor discharge valves, In *Proceedings of the International Compressor Engineering Conference at Purdue*, Paper 1112. West Lafayette-IN, EUA.

Kinjo, K., Nakano, A., Hikichi, T., Morinishi, K., (2010). Study of CFD considering valve behavior in reciprocating compressor, In *Proceedings of the International Compressor Engineering Conference at Purdue*, Paper 1256. West Lafayette-IN, EUA.

Mayer, J., Bjerre, P., Brune, F., (2014). A comparative study of different numerical models for flapper valve motion. In *Proceedings of the 22nd International Compressor Engineering Conference at Purdue*, Paper 1315. West Lafayette-IN, EUA.

Mistry, H., Bhakta, A., Dhar, S., Bahadur, V., Dey, S., (2012). Capturing valve dynamics in reciprocating compressors through computational fluid dynamics, In *Proceedings of the International Compressor Engineering Conference at Purdue*, Paper 1210. West Lafayette-IN, EUA.

Possamai, F. C., Todescat, M. L., (2004). A review of household compressor energy, In *Proceedings of the International Compressor Engineering Conference at Purdue*, Paper 1711. West Lafayette-IN, EUA.

Silva, J. C., Arceno, E., (2014). Correlation between the fluid structure interaction method and experimental analysis of bending stress of a variable capacity compressor suction valve. *In Proceedings of the 22nd International Compressor Engineering Conference at Purdue*, Paper 1324. West Lafayette-IN, EUA

Tan, Q., Liu, Z., Cheng, J., Feng, Q., (2014). Effective flow and force areas of discharge valve in a rotary compressor. *In Proceedings of the 22nd International Compressor Engineering Conference at Purdue*, Paper 1159. West Lafayette-IN, EUA.

Wu, S., Wang, Z., (2014). A numerical simulation of fluid-structure interaction for refrigerator compressors suction and exhaust system performance analysis, *In Proceedings of the International Compressor Engineering Conference at Purdue*, Paper 1131. West Lafayette-IN, EUA.

Yoshizumi, F., Kondoh, Y., Moroi, T., Tamano, S., Morinishi, Y., (2014). Numerical simulation on the opening delay of a discharge valve in compressor. *In Proceedings of the 22nd International Compressor Engineering Conference at Purdue*, Paper 1428. West Lafayette-IN, EUA.

ACKNOWLEDGEMENT

The authors acknowledge the support provided by FAPESP (Grant n° 2010/16837-9), TECUMSEH do Brasil, and FUNDUNESP (Grant n° 01525/11).